Scientific Journal of Silesian University of Technology. Series Transport
Zeszyty Naukowe Politechniki Śląskiej. Seria Transport

Volume 91 **2016**

p-ISSN: 0209-3324

e-ISSN: 2450-1549

DOI: 10.20858/sjsutst.2016.91.12

Journal homepage: http://sjsutst.polsl.pl

Article citation information:
Przemyk, A., Harlecki, A., Gancarczyk, T. Application of MSC Adams – NX Nastran/Femap interface in strength calculations of truck frames. *Scientific Journal of Silesian University of Technology. Series Transport*. 2016, **91**, 113-123. ISSN: 0209-3324.
DOI: 10.20858/sjsutst.2016.91.12.

Adam PRZEMYK[1], Andrzej HARLECKI[2], Tomasz GANCARCZYK[3]

APPLICATION OF MSC ADAMS – NX NASTRAN/FEMAP INTERFACE IN STRENGTH CALCULATIONS OF TRUCK FRAMES

Summary. In this paper, the finite element method (FEM) is used to calculate the strength of truck frames by integrating the MSC Adams software, for dynamics analysis of mechanical systems, and the NX Nastran/Femap software. At the same time, a method for reducing degrees of freedom is been developed based on the Craig–Bampton method. The interface is applied in order to calculate the strength of the frame in the selected truck, which runs on the test track. The selected model of truck can be treated as the virtual prototype that is useful in the design process.
Keywords: truck; frame; dynamics; FEM; MSC Adams; NX Nastran/Femap

1. INTRODUCTION

The development of computational techniques enables the construction of effective virtual prototypes of designed mechanical systems, which precede the construction stage of their real prototypes. Such a procedure significantly shortens the design process time, as well as

[1] Faculty of Mechanical Engineering and Computer Science, University of Bielsko-Biala, 2 Willowa Street, 43-309 Bielsko-Biala, Poland. E-mail: a.przemyk@gmail.com.
[2] Faculty of Mechanical Engineering and Computer Science, University of Bielsko-Biala, 2 Willowa Street, 43-309 Bielsko-Biala, Poland. E-mail: aharlecki@ath.bielsko.pl.
[3] Faculty of Mechanical Engineering and Computer Science, University of Bielsko-Biala, 2 Willowa Street, 43-309 Bielsko-Biala, Poland. E-mail: tgan@ath.bielsko.pl.

decreases its cost. The process of designing the mechanical systems, which are subjected to loads that are variable and difficult to determine, should be recognized as particularly complex. These systems include vehicles that are exposed to an impact of variable forces as a result of their motion over the uneven surfaces, which leads to frequently rapid manoeuvres, such as acceleration, braking, changing lanes or negotiating curves. Great importance is assigned to the design process of vehicle frames, particularly the frames of trucks, given that significant structures that must meet high safety requirements are involved. Therefore, such structures should not only be characterized by appropriately high ultimate strength, but also by fatigue strength, because they are exposed to dynamic changes in their loads, which cause vibrations. However, due to the fact that there is a necessity to maintain appropriately high performance characteristics in a vehicle, the frame weight should be limited. Strength calculations of truck frames, which are made within their design process, are generally based on the formal use of the FEM. In recent years, numerous studies have been devoted to issues involving strength calculations of truck frames using the FEM. Their authors have used different computing environments. A list of works dealing with the strength analysis of truck frames, which frequently use the ANSYS program, is presented in [10]. Meanwhile, the authors in [5, 6, 8] use the Nastran program, which is gaining more and more recognition among design engineers, whereas the authors in [4] used the Abaqus package. An application of the LS-DYNA® program, for the strength analysis of truck frames subjected to crash tests, was described in [9], while an application of this program described in article [5] regarding the strength analysis of a frame of a semi-trailer tractor moving over an uneven surface. Strength calculations of truck frames using the FEM are performed for static and dynamic loads in both the linear and non-linear ranges. The dynamic calculations should not be limited to the determination of ultimate stresses. Fatigue calculations should be fundamental here – for example, this issue was dealt with by the authors in [1, 5, 6]. On the whole, FEM calculations are not limited to determining stresses and displacements, because a modal analysis, which enables the determination of natural frequencies and mode shapes of free vibrations of the frames, can also have a great significance. It is important that these frequencies do not overlap with frequencies, which emanate from an engine or are the result of road surface unevenness. For example, results of the modal analysis of such frames were described in [3, 7].

2. GEOMETRIC MODEL OF A VEHICLE

The object under analysis in this work is a frame of the selected truck, whose geometric model, made by using the SolidWorks program, is presented in Fig. 1. The geometric model of the vehicle involves the assembly of the geometric models of its subsystems, as specified in Table 1, including the frame (Fig. 2), which is its fundamental part.

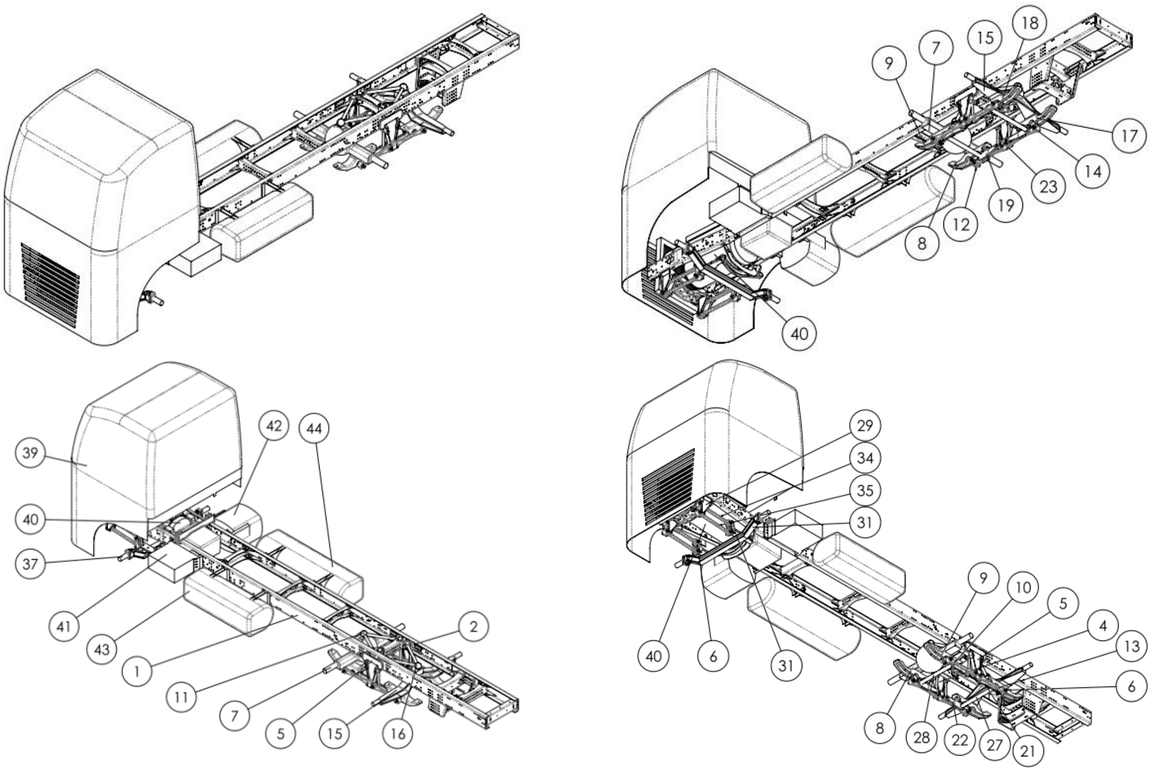


Fig. 1. A geometric model of the vehicle

Table 1. A list of the main elements of the geometric model of the vehicle

Element number	Description
1	Frame
2, 3	Internal reinforcements of the frame
4, 5	Right and left bracket of second and third axle suspension
6	Connector
7	Second axle with a differential
8, 9	Yoke of second axle
10, 12	Lower control arms of the second axle
11	Upper control arm of the second axle
13, 14	Lower control arms of the third axle
15	Third axle
16	Upper control arm of the third axle
17, 18	Yoke of the third axle
19, 21, 22, 27, 28	Elements of the stabilizer unit
23	Stabilizer link of the second axle
29	Left bracket of the first axle suspension

31	First axle
34, 35	Control arms of the first axle
37	Steering knuckle of the first axle
39	Cab
40	Power train (engine with equipment, clutch, gearbox)
41	Air tanks and battery box
42	Silencer with exhaust gas after-treatment system
43, 44	Fuel tanks

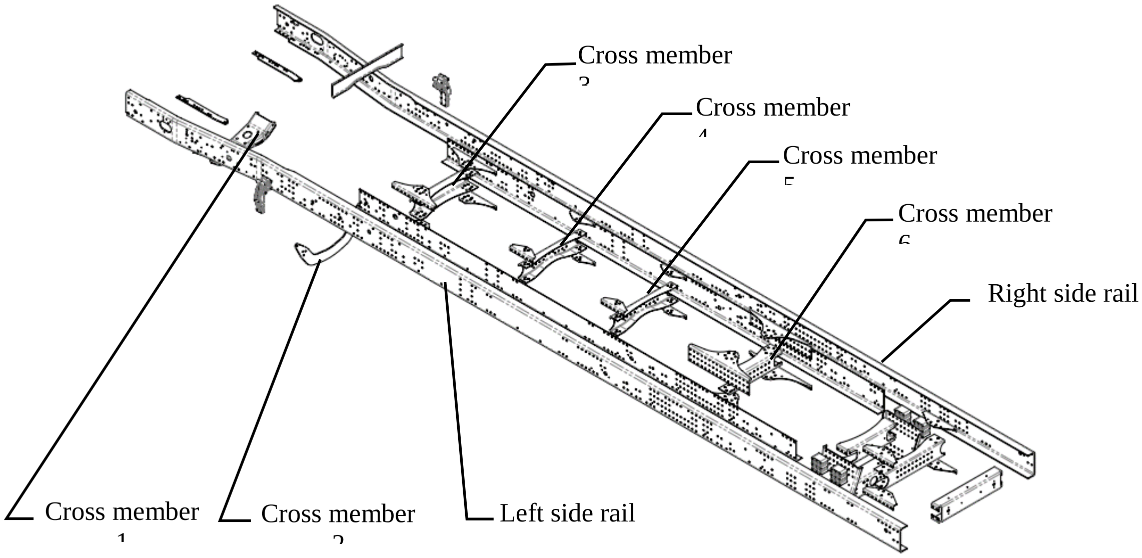


Fig. 2. A geometric model of the frame – a view of particular elements

3. FEM OF THE FRAME

An FEM of the frame was built on the basis of its geometric model by using the NX Nastran/Femap environment. This model contains around 148,000 finite elements, which are mainly shell (QUAD8, QUAD4, TRIA6, TRIA3) and solid (TETRA10) varieties, 383,000 nodes and a number of its degrees of freedom, which equates to 2,300,000.

In the places where the MSC Adams model of the analysed vehicle was found, the frame was connected with components adjacent to it, while additional nodes were generated. Those nodes were joined with several FEM grid nodes, which were adjacent to them, by rigid bar elements RBE2.

4. MSC ADAMS VEHICLE MODEL

An MSC Adams model of the analysed vehicle is presented in Fig. 3, while Table 2 lists its components.

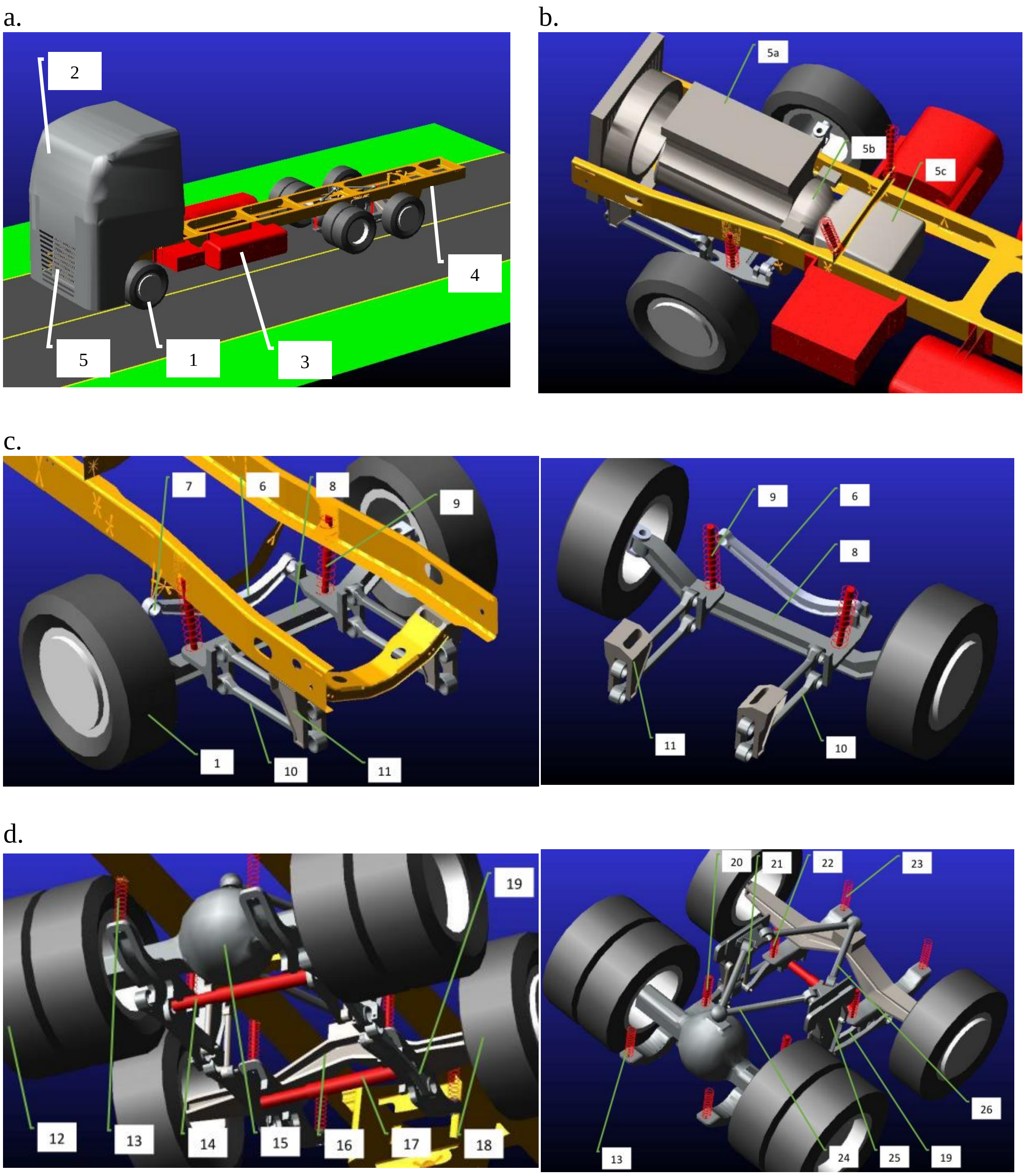


Fig. 3. An MSC Adams vehicle model: a) general view, b) power train, c) front suspension and d) rear suspension

Tab. 2

A list of the main components of an MSC Adams vehicle model

Element number	Description
1	Wheels of first axle
2	Cab
3	Fuel tanks, air tanks and battery box
4	Frame
5	Power train (a – engine, b – clutch, c – gear box)
6	Panhard rod
7	Place where Panhard rod is connected with frame
8	First axle
9	Air spring bellows and damper of first axle suspension
10	Control arm of the first axle suspension
11	Right bracket of the first axle suspension
12	Wheels of the second axle
13	Front air spring bellows of the second axle suspension
14	Elements of the stabilizer unit of the second axle
15	Second axle
16	Third axle
17	Elements of the stabilizer unit of the third axle
18	Wheels of the third axle
19	Lower control arm of the third axle
20	Rear air spring bellows and damper of the second axle suspension
21	Stabilizer link of the third axle
22	Front air spring bellows of the third axle suspension
23	Rear air spring bellows and damper of the third axle suspension
24	Upper control arm of the second axle
25	Left bracket of the second and third axle suspensions
26	Upper control arm of the third axle

Parameters of the wheel tyres were defined on the basis of a model of the PAC2002 tyre, as offered by the MSC Adams program, as well as the “Magic Formula” tyre model, as developed by Pacejka and Bakker [11].

5. INTERFACE APPLICATION BETWEEN TMSC ADAMS AND FEM PROGRAMS

The transfer of information between particular programs, as realized within the interface between the MSC Adams and NX Nastran/Femap programs, is presented in Fig. 4. A method for reducing the degrees of freedom is also developed here, based on the Craig–Bampton approach [2].

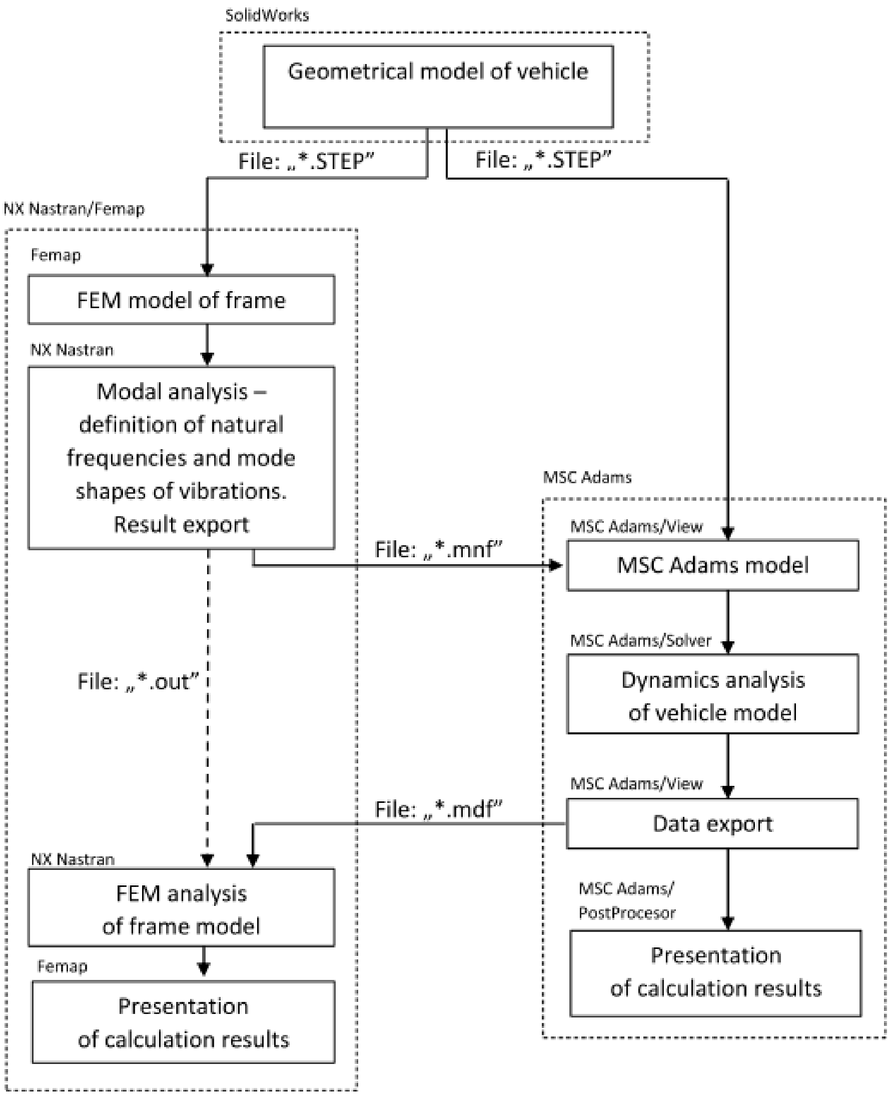


Fig. 4. Interface between the MSC Adams and NX Nastran/Femap programs – information transfer

6. SOME CALCULATION RESULTS

Within the computing tests, different cases of vehicle motion over smooth and uneven road surfaces were analysed. Some calculation results, regarding the simulation of a vehicle that drives with left wheels over a 50mm–high obstacle in the form of a bump with a rectangular cross-section, are presented in successive figures.

Time courses for the forces of the suspension spring and damping elements, which act on the frame, are presented in Fig. 5.

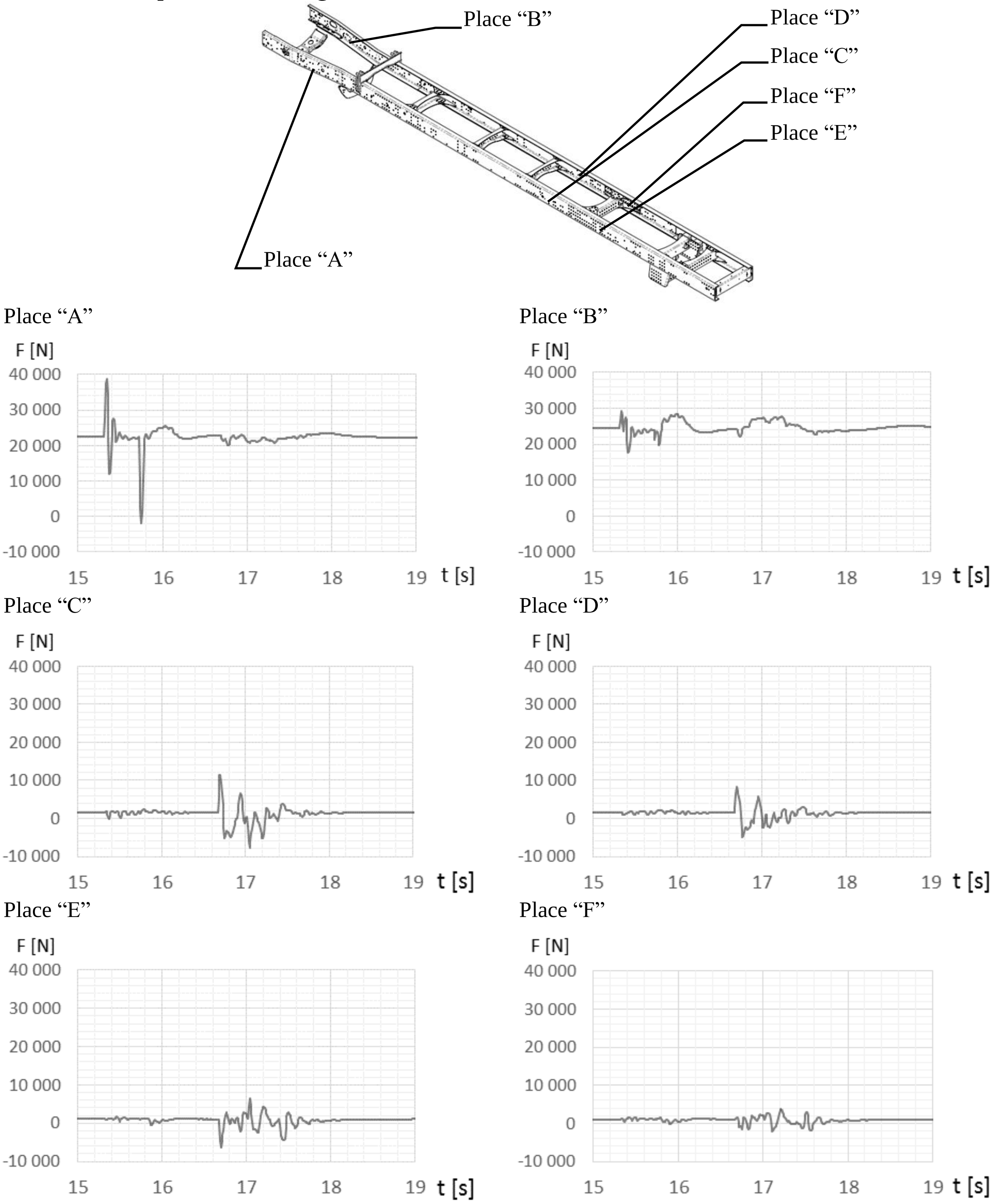
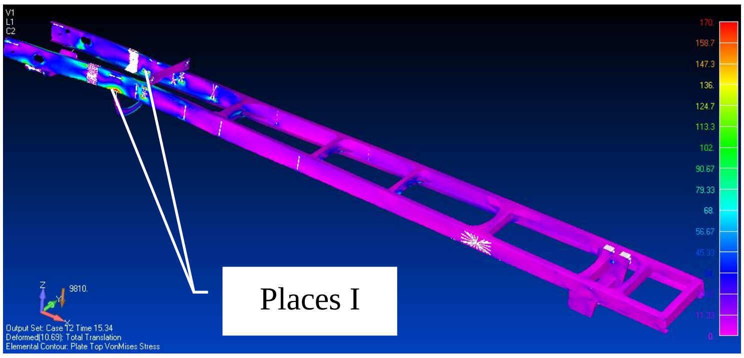


Fig. 5. Courses of suspension forces regarding the spring and damping elements acting on the frame

As expected, the forces of a higher value interact with the left part of the frame. An increase of forces in the places marked “A”, “B” and “C” occurs when the subsequent wheels drive over the obstacle. A particularly high force is present in place “A”, where there is also the highest static load of the frame, which results from the weight of the subsystems mounted on it.

Contours of the Huber–von Mises equivalent stresses in the frame, in which the force impacting on it reaches the maximum value in place “A” (where the left wheel of the axle drives towards the obstacle), are presented in Fig. 6a. The maximum values of the stresses in the cross members of the frame (Fig. 2) do not exceed 100MPa, whereas the stresses in the side rails of the frame reach local values of around 170MPa. The highest stresses occur in place I in Fig, 6a, where cross member 2 is mounted on the side rails of the frame, which is an area that is laden with the mounted gearbox. Contours of the Huber–von Mises equivalent stresses, in which the force impacting on the frame in place “C” reaches the maximum value (where the left wheel of the second axle drives towards the obstacle), are presented in Fig. 6b. In turn, the stresses in the cross members of the frame reach the value of around 170MPa, whereas the stresses in the left side rail reach the value of around 120MPa. Meanwhile, in the area where this side rail is particularly loaded as a result of its bending in two planes (place II in Fig. 6b), the value of these stresses reaches 170MPa.

a.



b.

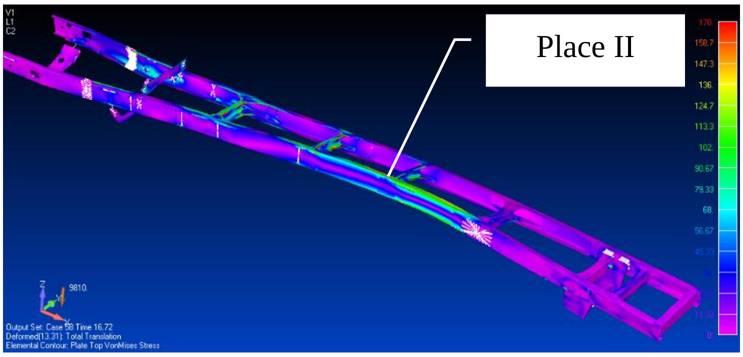


Fig. 6. Contours of the Huber–von Mises equivalent stresses in the frame:
a) at the moment of 15.34 s and b) at the moment of 16.72 s

The computational tool used in this study, which involves the interface between the MSC Adams and NX Nastran/Femap computer programs, allows for any computer simulations of the vehicle motion in question to be considered. However, the obtained results should only be treated as indicative. Their correctness should be confirmed by performing a series of experimental studies, which, for example, deal with the determination of real stresses in the vehicle frame by tensiometric measurements.

7. CONCLUSIONS

According to the authors, the method presented in this paper may be of interest to engineers dealing with the design of truck frames. The computer model of a vehicle that was developed with the use of an interface between the MSC Adams and NX Nastran/Femap programs, which was conceived as a virtual prototype, enables any set of calculations to be performed during the design process, such that the results ought to provide an image relating to the loads of its subsystems close to the real image.

References

1. Chetan S.J., C.P. Khushbu, P. Fajalhusen. 2012. „A review of fatigue analysis of automobile frames." *International Journal of Advanced Computer Research* 2 (4): 103-107. ISSN (online): 2277-7970.
2. Craig R. R., M. C. Bampton. 1968. „Coupling of substructures for dynamic analyses". *AIAA Journal* 6 (7): 1313-1319. ISSN 0001-1452.
3. Da Silva M.M., L.P.R. De Oliveira, L.G.S. Ericcson, A.C. Neto, P.S. Varoto. 2003. „An experimental investigation on the modal characteristics of an off-road competition vehicle chassis". *SAE Technical Paper* 2003-01-3689.
4. Dębski H., G. Koszałka, M. Ferdynus. 2012. „Application of FEM in the analysis of the structure of a trailer supporting frame with variable operation parameters". *Eksploatacja i Niezawodnosc - Maintenance and Reliability* 14 (2): 107-113. ISSN: 1507-2711.
5. Edara R., S. Shih, N. Tamini, T. Palmer, A. Tang. 2008. „18-wheel truck dynamic and durability analysis using virtual proving ground". In *10th International LS-DYNA® User Conference*. Detroit, USA. ISBN: 0-9778540-4-3.
6. Fischer P., W. Witteveen. 2000. „Integrated MBS – FE – durability analysis of truck frame components by modal stresses". In *ADAMS User Meeting*. Rome, Italy.
7. Fui T. H., R. A. Rahman. 2007. „Statics and dynamics structural analysis of 4.5 ton truck chassis." *Jurnal Mekanikal* (24): 56-67. ISSN (online): 2289-3873.
8. Koike M., S. Shimoda, T. Shibuya, H. Miwa. 2004. "Development of kinematical analysis method for vehicle". *Komatsu Technical Report* 50 (153).
9. Liu Y. 2010. „Crashworthiness analysis of finite element truck chassis model using LS-DYNA®". In *11th International LS-DYNA® User Conference*. Detroit, USA. ISBN: 0-9778540-5-1.
10. Moaaz A. O., N. M. Ghazaly. 2014. „Finite element stress analysis of truck chassis using ANSYS: review". *International Journal of Advances in Engineering & Technology* 7 (5): 1386-1391. ISSN: 22311963.

11. Pacejka H.B., E. Bakker. 1992. „The magic formula tyre model”. *Vehicle System Dynamics* 21: 1-18. ISSN: 0042-3114.

Received 20.12.2015; accepted in revised form 06.05.2016